
Numerical Simulation of Viscous Supersonic Flow Over a Generic Fighter Configuration

Denny S. Chaussee, Gordon Blom, and John Wai

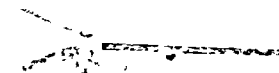
December 1985

LIBRARY COPY

JAN 23 1986

LANGLEY RESEARCH CENTER
LIBRARY, NASA
HAMPTON, VIRGINIA

FOR REFERENCE



NOT TO BE TAKEN FROM THE ROOM



National Aeronautics and
Space Administration



NF00066

Numerical Simulation of Viscous Supersonic Flow Over a Generic Fighter Configuration

Denny S. Chaussee, Ames Research Center, Moffett Field, California
Gordon Blom,
John Wai, Boeing Company, Seattle, Washington

December 1985



National Aeronautics and
Space Administration

Ames Research Center
Moffett Field California 94035

N86-15627#

NUMERICAL SIMULATION OF VISCOUS SUPERSONIC FLOW OVER A GENERIC FIGHTER CONFIGURATION

Denny S. Chaussee
NASA Ames Research Center
Moffett Field, CA 94035 USA

Gordon Blom and John Wai
Boeing Company
Seattle, WA 98124 USA

INTRODUCTION

In the next generation of fighter aircraft, the mission requirements dictate a versatile vehicle that can operate over a wide range of conditions. One requirement is likely to be a supersonic cruise capability. In addition, the vehicle will need to be able to maintain its maneuverability as well as its low-speed characteristics. To provide this capability in such an aircraft, devices such as close-coupled canards and vortex control are being considered.

In an effort to address the impact of these technologies, a computational tool is being developed by NASA Ames Research Center and the Boeing Military Airplane Company (BMAC). An example of the aircraft being considered for this development is shown in Fig. 1. It is designed to cruise at Mach 2. Since there has been extensive wind tunnel testing of this configuration [1], sufficient data exist for verification of the computer code. Thus, the basic intent of this investigation is to develop a computational fluid dynamics (CFD) tool that is capable of analyzing the viscous supersonic flow about these realistic configurations. To meet this challenge, a technique is being developed to predict the entire flow field about the configuration, including difficult flow regions near canopy, wing, and canards. The prediction technique must yield a physically meaningful solution when flow phenomena such as the interaction of the canard vortex and wing occur. Vortex bursting, which causes both unsteady flow and axial separated flow, is likely to occur in such a configuration and invalidate the numerical marching procedure. In such instances, a time-dependent Navier-Stokes code will be used to solve the problem of the interacting flows over the wings.

In this paper, the only results presented are those for the supersonic flow over the wing-canopy body shown in Fig. 1. A viscous numerical-space marching procedure (the NASA Ames Parabolized Navier-Stokes (PNS) code [2-5]) was employed to compute these flows. The flow conditions simulate supersonic cruise with a freestream Mach number of 2.169 and angles of attack of 4° and 10° . The body surface was considered to be an adiabatic wall and the flow was assumed to be turbulent for the given Reynolds number.

COMPUTATIONAL TECHNIQUE

The PNS equations are obtained from the complete Navier-Stokes equations by neglecting the unsteady terms and the streamwise-viscous derivative terms (see [2]).

In the present formulation, ξ (the marching direction in computational space) is a function of X only (this implies axis-normal marching planes). The governing equations are hyperbolic-parabolic in the ξ -direction if the inviscid part of the flow field is supersonic, if there is no streamwise (axial) separation, and if the pressure gradient in the viscous region near the wall is treated correctly [2]. However, the PNS system of equations still permits separation in the crossflow plane ($\eta - \zeta$).

The present PNS code uses the Beam-Warming implicit algorithm to solve the parabolized approximation to the Navier-Stokes equations and characteristic, implicit, spatially second-order-accurate boundary conditions at the outermost shock wave. An elliptic grid generator of the type developed by Steger and Sorenson [6], which is further specialized to wing bodies by Rai et al. [3], is used to generate the grid for the calculations. The algebraic turbulence model was developed by Baldwin and Lomax [7]. At the body surface, the viscous no-slip boundary condition is applied. Since the equations are cast in conservation-law form, all discontinuities within the flow domain are predicted correctly.

To initiate the marching procedure, a starting plane (or planes) of data is required. This initial solution is obtained as follows: A shock location is obtained from the configuration and flow conditions, and the flow properties on the surface are obtained from NACA 1135 for a region near the nose of the vehicle. These properties are used to estimate the flow in the shock layer. Since this estimated flow field was obtained very near the nose tip, a viscous solution can be generated as the PNS code marches downstream by using very small marching steps. If this procedure is followed, a reasonable starting plane is obtained without expending a large amount of computer time. Also, the remaining flow field over the vehicle is largely unaffected because it is near the nose.

RESULTS

Numerical results for supersonic cruise at $M_\infty = 2.169$ are presented. The wind tunnel conditions considered are such that the Reynolds number is turbulent. Adiabatic wall conditions are assumed at the body's surface. Angles of attack of 4° and 10° are considered. The absence of yaw results in a pitch plane of symmetry which reduces the computational space and hence the amount of central processing unit (CPU) time and storage requirements. The current geometry, consisting of a canopy, body, and wing, is in a form that is readily usable by the PNS code. The point distribution on each cross section was determined independently and was then used to produce a geometry file which was read into the PNS code. Intermediate cross sections were determined by simple interpolation in the axial direction. All of the figures are scaled by either the model length (L), [Fig. 1], or the local span width (b). The locations of some of the referenced cross sections are shown in Fig. 1.

A typical grid obtained by using the elliptic mesh generator is presented in Fig. 2. The outermost boundary is the location of the fitted bow shock. This particular grid has 61 meridional points and 45 radial points.

Figures 3 and 4 are mesh studies for two different X/L stations. Each study used 45 radial points, but varied the number of meridional points: 61, 91, and 121 for $\alpha = 4^\circ$. The results were compared with experimental results. The coarse details of the flow field were reasonably resolved by

all systems; however, for the $X/L = 0.8$ station it appears that the 121 X 45 grid system produces the more accurate overall results.

Comparisons between experimental and computational values of C_p at $\alpha = 10^\circ$ for different X-stations are presented in Figs. 5, 6, and 7. The agreement at all of the X-stations, -- $X/L = 0.575, 0.71, \text{ and } 0.8$ -- is very good. The distribution of grid points on the body surface at each of these stations is represented by the symbols on each cross-sectional plot.

In Fig. 8, a comparison between the experiment and computational pitot pressures at three meridional locations for an $X/L = 0.65$ is shown. The comparison shows very good agreement through the shock layer for the experimental data surveys that were normal to the fuselage surface. There is a small difference for each of the surveys normal to the wing surface, which is most probably a result of the nonoptimal distribution of computational grid points in the vicinity of the wing-fuselage junction.

The pressure contours are presented in Fig. 9 at an $X/L = 0.672$ and $\alpha = 4^\circ$. The contours show a rapid expansion around the wingtip followed by a recompression.

The crossflow velocity vectors are presented for $\alpha = 10^\circ$ at an $X/L = 0.8$ in Fig. 10. Every third meridional value is plotted to enable an unobstructed view of the flow physics. Again, the rapid expansion around the leading edge is visible. At this higher angle of attack, a recirculation region exists on the upper surface of the wing. Along the leeward radial ray, the downstream effect of the expansion fan which was created by the canopy protruding into the oncoming flow is clearly visible. This is shown by the change in direction of the velocity vectors.

SUMMARY

Results are presented for supersonic flow about a wing-canopy body configuration. The thin-layer Navier-Stokes equations were marched over this configuration, yielding numerical results which agreed well with the experiment. This procedure is computationally efficient, reasonably robust, and can be a viable tool to investigate the aerodynamics and the fluid physics of supersonic flow past complex, winged configurations. In the future, the canard effects on the flow field will be studied by the present procedure or by a time-dependent procedure.

REFERENCES

1. Capone, F.J., Bare, E.A., Hollenback, D. and Hutchison, R., "Subsonic/Supersonic Aerodynamic Characteristics For a Tactical Supercruiser", AIAA Paper No. 84-2192, 1984
2. Schiff, L.B. and Steger, J.L., "Numerical Simulation of Steady Supersonic Viscous Flow", AIAA Paper No. 79-0130, 1979
3. Rai, M.M. and Chaussee, D.S., "New Implicit Boundary Procedures: Theory and Applications", AIAA Paper No. 83-0123, 1983
4. Rai, M.M., Chaussee, D.S. and Rizk, Y.M., "Calculations of Viscous Supersonic Flows Over Finned Bodies", AIAA Paper No. 83-1667, 1983

5. Chaussee, D.S., Rizk, Y.M. and Buning, P.G., "Viscous Computations of a Space Shuttle Flow Field", Lecture Notes in Physics: Ninth International Conference On Numerical Methods In Fluid Dynamics, 218, 1984, pp. 148-153

6. Steger, J.L. and Sorenson, R.L., "Automatic Mesh-Point Clustering Near a Boundary In Grid Generation With Elliptic Partial Differential Equations", J. Of Computational Physics, Vol. 33, 1979, pp. 405-410

7. Baldwin, B.S. and Lomax, H., "Thin Layer Approximation and Algebraic Model For Separated Turbulent Flow", AIAA Paper No. 78-257, 1978

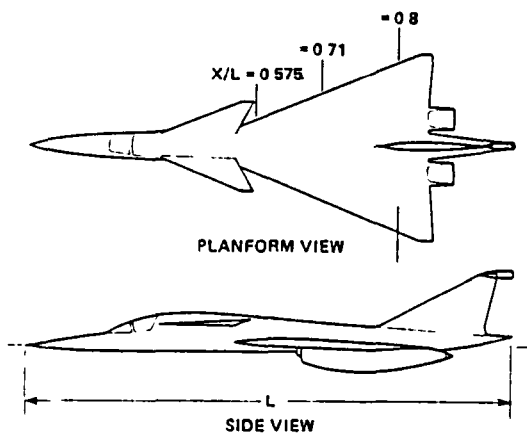


Fig 1 Geometry of a generic supercruise fighter

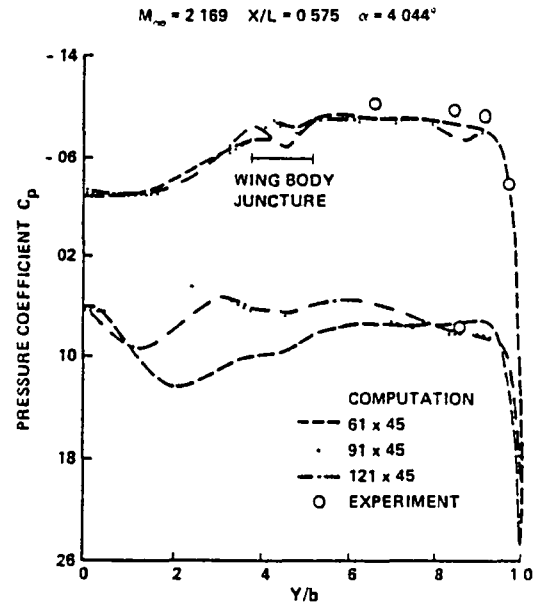


Fig 3 Mesh effects at $X/L = 0.575$ and $\alpha = 4^\circ$

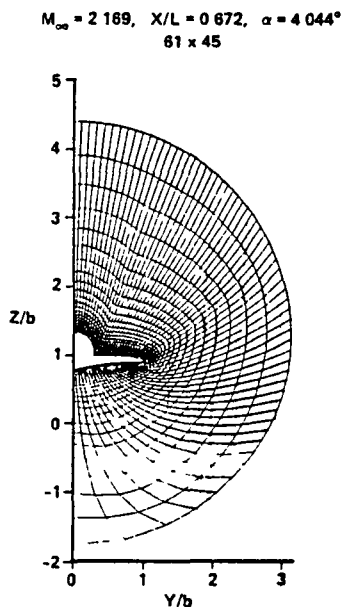


Fig 2 Elliptic grid at $X/L = 0.672$ and $\alpha = 4^\circ$

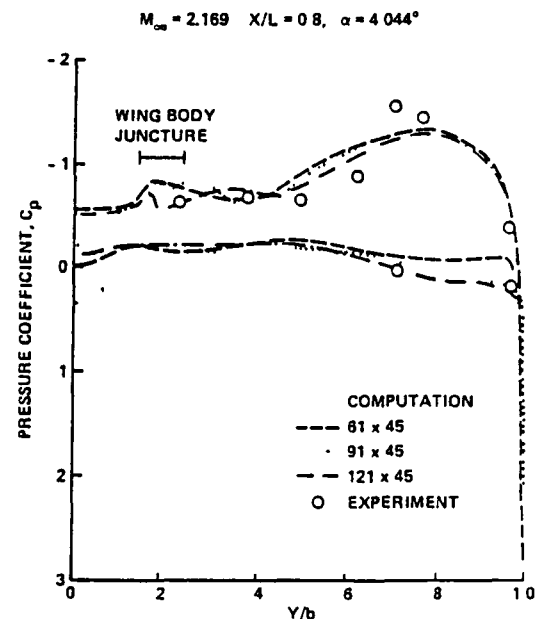


Fig 4 Mesh effects at $X/L = 0.8$ and $\alpha = 4^\circ$

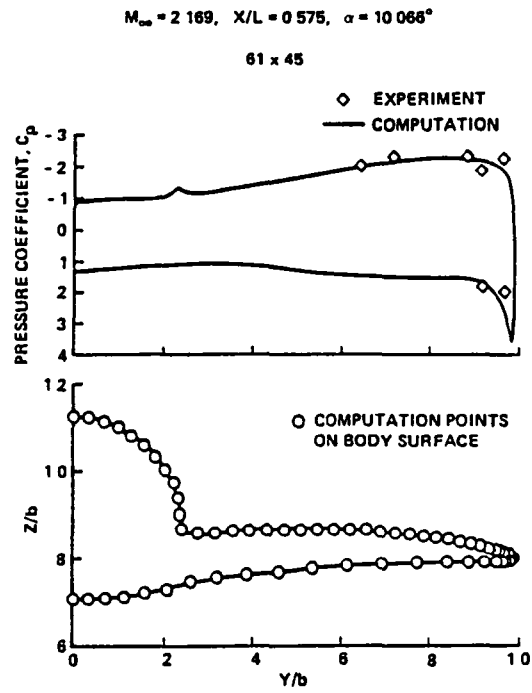


Fig. 5 Pressure coefficient comparisons at $X/L = 0.575$ and $\alpha = 10^\circ$ for a 61-point grid

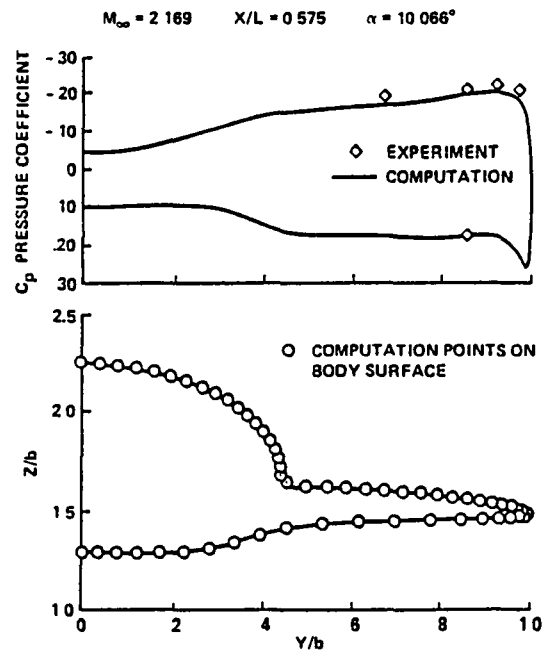


Fig. 6 Pressure coefficient comparisons at $X/L = 0.710$ and $\alpha = 10^\circ$ for a 61-point grid

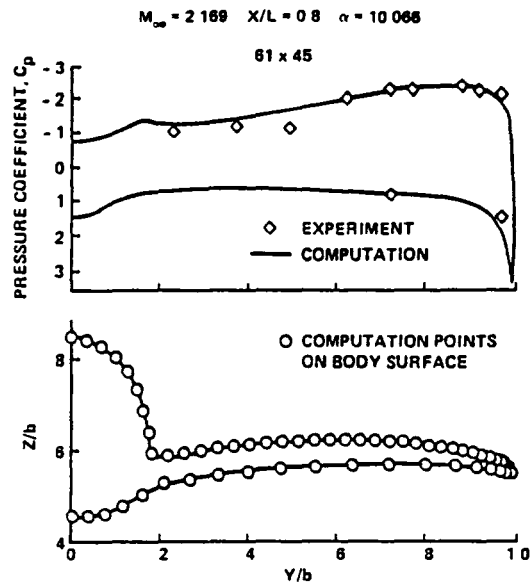


Fig. 7 Pressure coefficient comparisons at $X/L = 0.800$ and $\alpha = 10^\circ$ for a 61-point grid

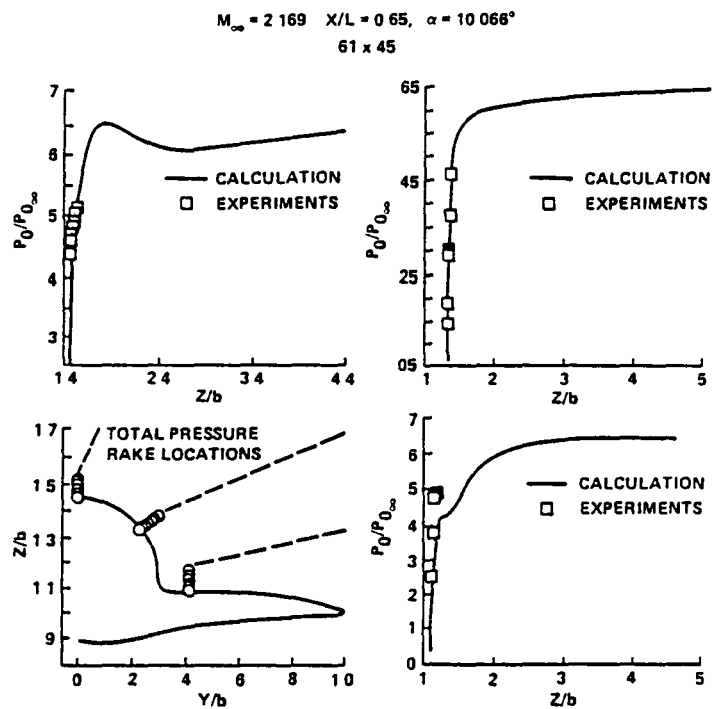


Fig. 8 Pitot pressure comparisons for various meridional stations at an $X/L = 0.65$ and $\alpha = 10^\circ$

61 x 45



61-point grid

91 x 45



91-point grid

1 Report No NASA TM-86823	2 Government Accession No	3 Recipient's Catalog No	
4 Title and Subtitle NUMERICAL SIMULATION OF VISCOUS SUPERSONIC FLOW OVER A GENERIC FIGHTER CONFIGURATION		5 Report Date December 1985	
		6 Performing Organization Code	
7 Author(s) Denny S. Chaussee*, Gordon Blum†, and John Wai†		8 Performing Organization Report No A-85436	
		10 Work Unit No	
9 Performing Organization Name and Address *Ames Research Center, Moffett Field, CA 94035 †Boeing Company, Seattle, WA 98125		11 Contract or Grant No	
		13 Type of Report and Period Covered Technical Memorandum	
12 Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546		14 Sponsoring Agency Code 505-31-01	
15 Supplementary Notes Point of Contact: Denny S. Chaussee, Ames Research Center, MS 202-14, Moffett Field, CA 94035, (415) 694-6742 or FTS 464-6742			
16 Abstract A procedure is presented, as well as some results, to calculate the flow over a generic fighter configuration. A parabolized marching Navier-Stokes code is used to obtain the solution over a wing-canopy body. The flow conditions simulate supersonic cruise with a freestream Mach number of 2.169 and angles of attack of 4° and 10°. The body surface was considered to be an adiabatic wall and the flow was assumed to be turbulent for the given Reynolds number.			
17 Key Words (Suggested by Author(s)) Computational fluid dynamics Supersonic flow Three-dimensional viscous flow Supercruise		18 Distribution Statement Unlimited Subject category: 34	
19 Security Classif (of this report) Unclassified	20 Security Classif (of this page) Unclassified	21 No of Pages 8	22 Price* A02

End of Document